MYSTRAN

General Purpose Finite Element Structural Analysis

Computer Program

by Dr Bill Case

MYSTRAN Overview:

- **MYSTRAN is an acronym for My Structural Analysis.**
- **Static and eigenvalue analyses of linear structures.**
- **Computer program for PC's that is a "NASTRAN" look-alike:**
  - **Input data deck entries are the same format as NASTRAN.**
  - **Outputs look like NASTRAN.**
  - **Many current NASTRAN input data decks will run with no modification on MYSTRAN.**
  - **Problem size limited only by resources on your personal computer.**
- **About the computer code:**
  - **All of the finite element processing to obtain the global stiffness matrix (including the finite element matrix generation routines), the reduction of the stiffness and mass matrices to the solution set, as well as all of the input/output routines are written in independent, modern, Fortran 90/95 code (*i.e. MYSTRAN code is not NASTRAN code*).**
  - **The major solution algorithms (e.g., triangular decomposition of matrices and forward/backward substitution to obtain solutions of linear equations as well as eigenvalue extraction routines) were obtained from the popular LAPACK code available to the general public on the World Wide Web.**

Major features of the program are:

- **Linear static analysis.**
- **Eigenvalue analysis.**
- **3D structures with arbitrary geometry.**
- **Grid points of the finite element model that can be defined in arbitrary rectangular, cylindrical and/or spherical coordinate systems. In addition, the global stiffness matrix can be formulated, and the displacements calculated, in arbitrary coordinate systems for each grid that can be different than the one used to define the grid location.**
- **Six degrees of freedom (three translations and three rotations) per grid.**

(Major features continued on next 3 pages)

Major features (*continued*):

- **A finite element library consisting of the following elastic and rigid elements:**

  Elastic Elements:

  - **ELAS1 element (same as NASTRAN).**
  - **ROD element (same as NASTRAN).**
  - **BAR element (same as NASTRAN).**
  - **Triangular and quadrilateral plate elements for thick (Mindlin plate theory) *and* thin (Kirchoff plate theory) plates:**
    - **TRIA3 flat triangular plate element with plate membrane and bending stiffness as well as transverse shear flexibility.**
    - **QUAD4 quadrilateral plate element with plate membrane and bending stiffness, as well as transverse shear flexibility.**

  Rigid Element:

  - **RBE2 rigid element specifying a relationship for one or more degrees of freedom of one or more grids being rigidly dependent on the degrees of freedom of another grid.**

Major features (*continued*):

- **Single Point Constraints (SPC's) wherein specified degrees of freedom are grounded (e.g. for specifying boundary conditions).**
- **Other SPC's wherein specified degrees of freedom have a specified motion (enforced displacements).**
- **Multi point constraints (MPC's), wherein specified degrees of freedom are linearly dependent on other degrees of freedom.**
- **Loads on the finite element model via:**
  - **Forces and/or moments applied directly to grid points.**
  - **Pressure loading on plate element surfaces.**
  - **Gravity loads on the whole model (in conjunction with mass defined by the user).**
  - **Equivalent loads due to thermal expansion.**
  - **Equivalent loads due to enforced displacements.**
- **Linear isotropic material properties.**
- **Multiple subcases to allow for solution for more than one loading condition in one execution.**

Major features (*continued*):

- **Mass defined via:**
  - **Density on material entries.**
  - **Mass per unit length, or per unit area, for finite elements.**
  - **Concentrated masses at grids (CONM2) with offsets and moments of inertia.**
- **Guyan reduction to statically reduce the stiffness and mass matrices.**
- **Output of:**
  - **Displacements (6 degrees of freedom per grid) for any defined set of grids desired.**
  - **Applied loads for any defined set of grids.**
  - **Single point forces of constraint for any defined set of grids.**
  - **MPC and rigid element forces of constraint for any defined set of grids (next version released).**
  - **Grid point force balance for any defined set of grids (next version released).**
  - **Element engineering and/or nodal forces for any defined set of elements.**
  - **Element stresses for any defined set of elements.**

About the author:

Dr. Bill Case was a member of the project management team for the development of the popular NASTRAN finite element analysis computer program developed in the late 1960's and early 1970's for the National Aeronautics and Space Administration (NASA). Over the course of 30 plus years from that time he was a structural analyst using NASTRAN and subsequently the head of an analysis branch comprised of approximately 20 analysts who used NASTRAN on many of the spaceflight structures built or managed by NASA Goddard Space Flight Center (GSFC). He has taught many graduate and undergraduate level courses in the area of static and dynamic structural analysis as well as courses in the use of NASTRAN. He can be contacted via email at DrBillC@earthlink.net**.**

How to obtain a free trial copy of MYSTRAN:

Go to the web site http://www.MYSTRAN.com and follow the directions given on the "Download MYSTRAN" page to obtain a free trial version that can solve small problems. This version (1.02) can be "unlocked" to solve unlimited size problems via a "key" obtained from the author for a fee.

Next version of MYSTRAN will have the following improvements over the original version

- **Orthotropic material properties.**

- **Improved handling of plate offsets to allow different offset for any plate element via specification on the element connection entry instead of specification on plate property entries as in version 1.00 (currently in version 1.02).**

- **Improved storage requirements for matrices in eigenvalue analysis. Currently, in eigenvalue analyses, matrices are stored in full format. A banded matrix format will be used in future versions, saving disk memory. (Note: *this is only in eigenvalue analyses. In static analyses matrices are currently stored in a format that requires storage of only the nonzero terms*).**

- **Additional eigenvalue analysis techniques. Currently only a Givens (GIV) technique is available. Future versions will have modified Givens (MGIV) as well as a Lanczos technique.**

- **Correction of errors found in prior versions.**